PSPICE TUTORIAL

PART I: INTRODUCTION AND DC ANALYSIS

for the

Orcad PSpice Release 9.1 Student Version

INTRODUCTION

The Simulation Program with Integrated Circuit Emphasis (SPICE) circuit simulation tool was first developed in the early 1970s. It was written in the FORTRAN programming language and was intended to support the early data entry methods of this period. SPICE was immediately valuable to allow circuit designers to analyze circuit systems, in particular as the complexity of circuits began to expand with the arrival of the first integrated circuits. It is certainly one of the most important tools in Electrical Engineering and is an example of one of the first tools for Computer Aided Design.

SPICE has evolved with many advances in numerical analysis methods for accurate and fast computation and has appeared in many commercial forms. SPICE has been ported to many platforms and the version that operates over the Windows operating system is PSpice.

The first SPICE users designed circuits with manual circuit drawing tools. Then, inspection of the circuit design was used to generate a text-based description of the circuit design. Even today this still occurs in certain special instances. However, the arrival of graphical drawing tools allows a circuit designer to directly draw circuit schematics using circuit design tools and then special "capture" tools operate to "capture" the schematic and generate the text-based description of the circuit design, automatically. The versions of PSpice available to us operate in this way.

This is a great advantage for the engineering design process – this provides the designer with the ability to focus directly on a visual description of their system and obtain circuit operation numerical and graphical results in an automatic, convenient process.

This Tutorial describes the development of two typical, simple circuits. The first is a circuit that will demonstrate the ability to quickly compute voltage and current values. The second will demonstrate the ability to compute time-dependent circuit response.

ACCESS TO PSPICE

The SEASnet laboratory PCs all carry an installation of PSpice 9.2 Lite. If you wish to install this version of PSpice on your personal machine, you may order a free PSpice 9.2 Lite CD from Cadence at

http://www.orcad.com/Partner/Solution/ContentPage/cddemo4.asp

You may also download the PSpice 9.1 Student Version from

http://www.orcad.com/Product/Simulation/PSpice/download.asp

or just from the class web page:

http://www.ee.ucla.edu/~jjudy/classes/ee100/pspice/download/pspice_v9.1_student_version.zip

INSTALLATION

You may use the SEASnet PCs for your work. However, if you decide to install Orcad Family Release 9.2 Lite Edition or Release 9.1 Student Version software, then in the installation process you will be presented with options as to which components to install. Ensure that you have selected to install Capture and PSpice. You may install other options in addition to these.

Now, we will first use PSpice to simulate the circuit of Figure 2.16 in Electric Circuits in Nilsson and Riedel, shown below.

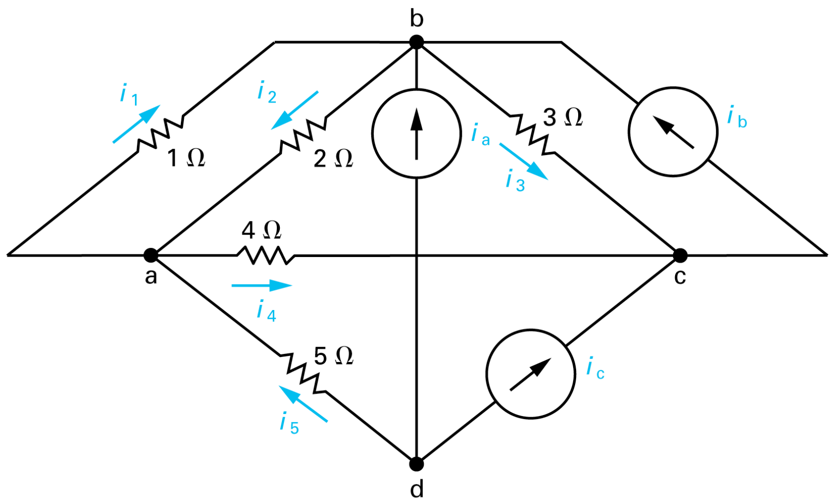

Figure 2.16 The circuit for Example 2.6

From: Nilsson/Riedel, Electric Circuits, 6e, July 2000 Prentice Hall, Inc.

Note: The book referred to above is the book used for EE 10. This tutorial has been adapted from the one used in EE 10, which is identical to the tutorial for PSpice 9.2 Lite.

GETTING STARTED

After installation, your Program folder should contain the PSpice software. So, select **Start > Programs > PSpice Student > Capture Student**. This will launch the Capture application.

Your screen should appear as in Figure 1 (except in v9.1 Student Version this log file will be minimized within the PSpice main window.

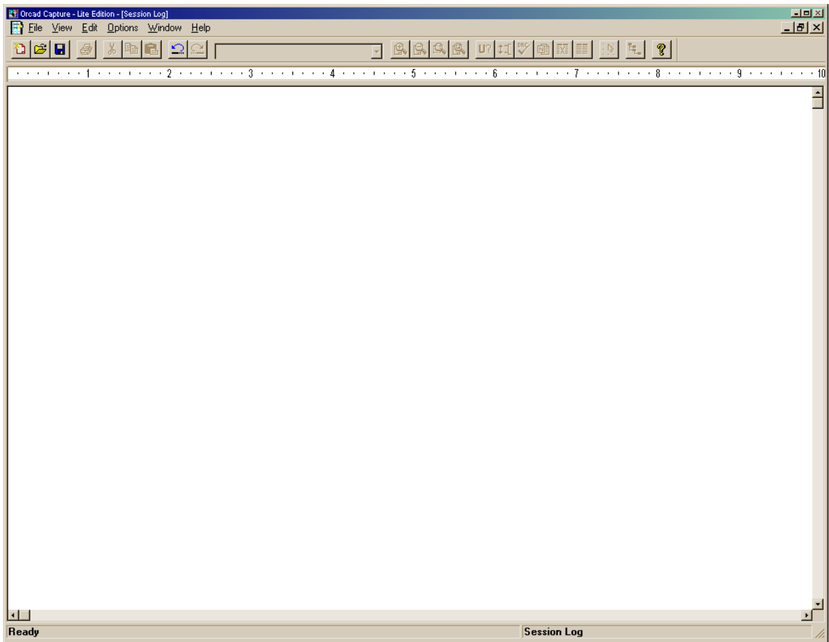

Figure 1

Now, to begin, you must create a new Project. For the purposes of this class, we will be engaged in Analog Design and thus your new Project selection will be for a new Analog Design project.

Begin by selecting the **File > New > Project** menu. Your screen should appear as in Figure 2.

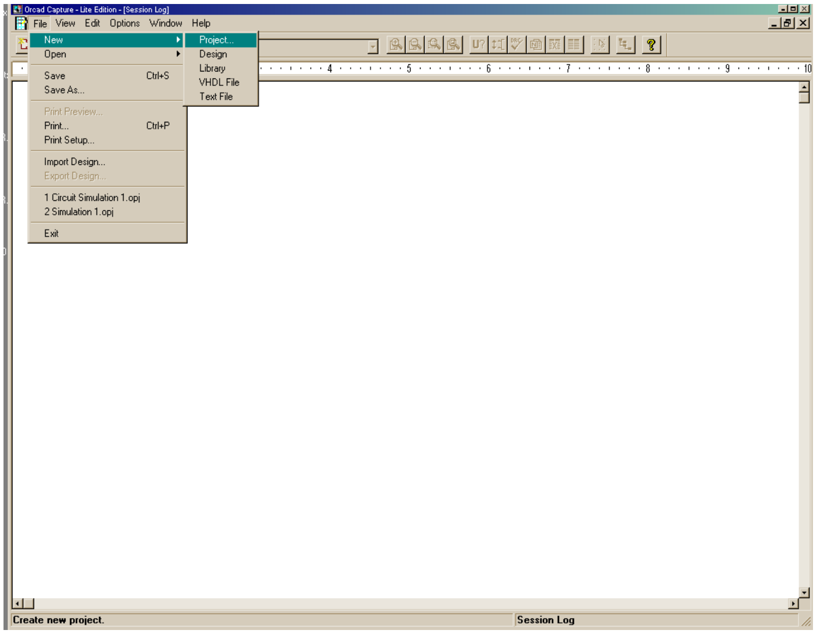

Figure 2

This will launch the Project dialog. At this point, you must **BROWSE** to create a directory for your new projects. Here we have created a directory of D:\My Documents\PSPICE\Projects in Figure 3.

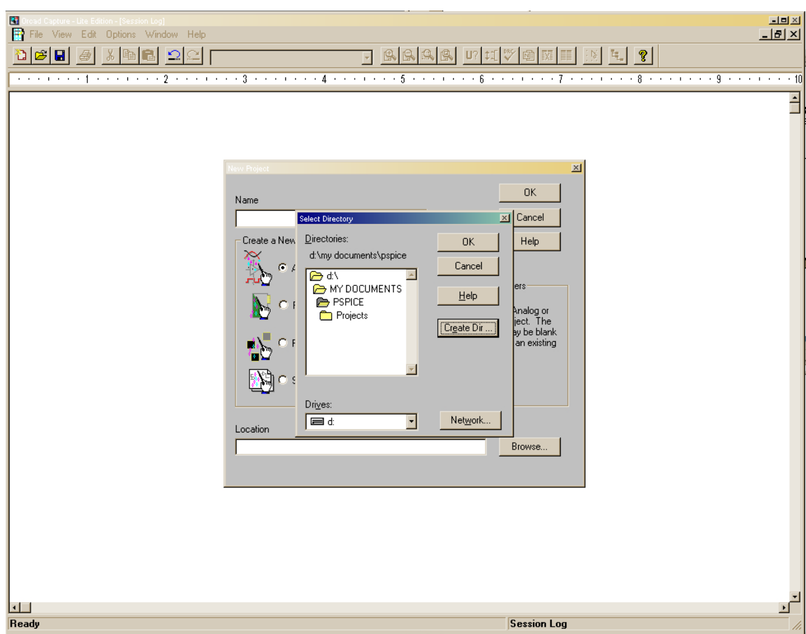

Figure 3

You should now configure the application so that an **Analog or Mixed A/D** (analog or mixed analog and digital design) is selected. Also, you should entire a Project Name. This has been selected to be, Figure 2.16 Circuit Simulation. Please note carefully that the Analog or Mixed A/D checkbox has been highlighted.

Now, upon confirming your selection by clicking on the OK button, another choice will appear. This prompts you as to whether you would like to create a new Project based on a previous Project. For this case, select **Create a blank project**, as shown in Figure 4 and then press the OK button.

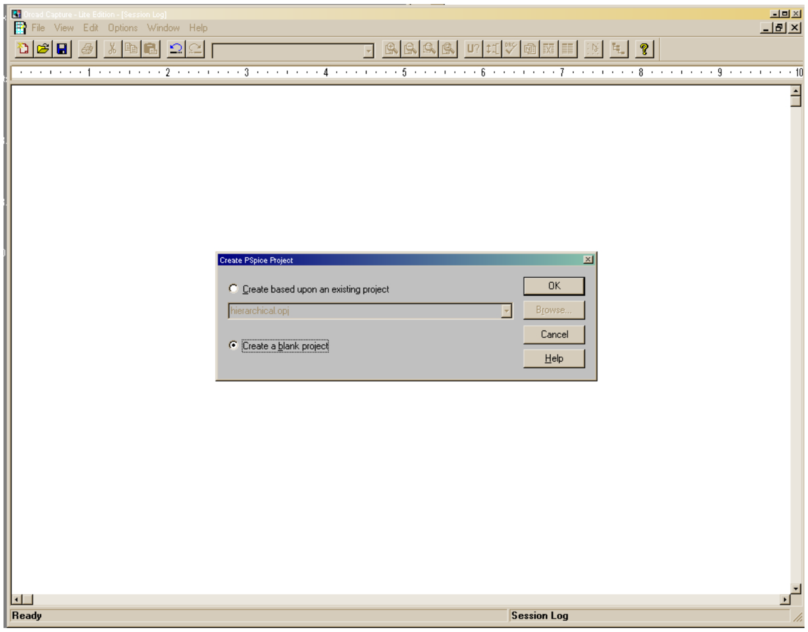

Figure 4

Now, this will bring up the Capture schematic graphics window, as in Figure 5. This will now allow you to actually draw a circuit with electronic components that can be individually adjusted for their properties. We are ready to begin designing.

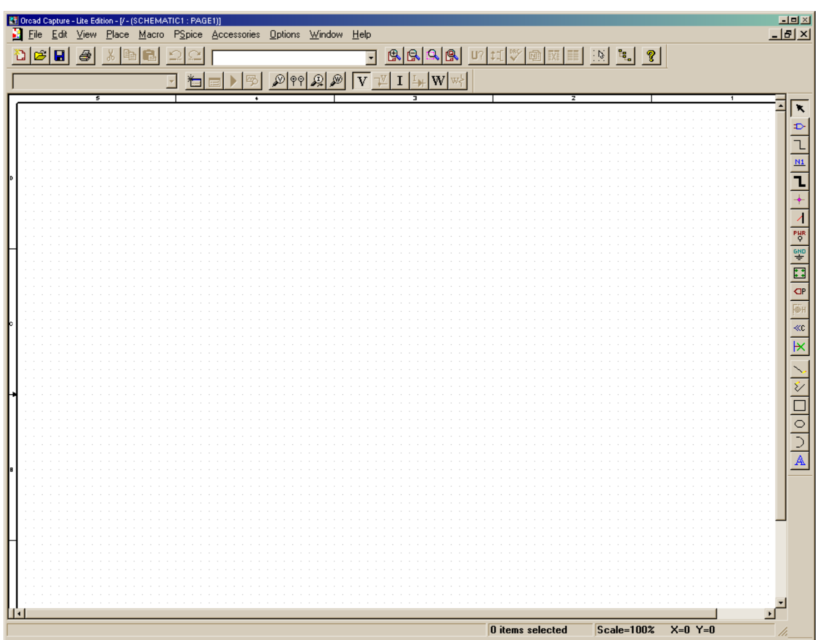

Figure 5

A very important point should be considered first: The PSpice system computes voltage values relative to a common potential (referred to as a “Ground” potential). Without a specification for this Ground potential, the circuit simulation may not proceed (the circuit simulation algorithm does not make assumptions as to what reference potential the designer has selected, instead, it must be informed in advance). In fact, the error condition resulting from a lack of Ground potential definition creates so-called “floating” nodes, simply circuit nodes that do not have a defined potential.

As you will see throughout engineering circuit design, the clear definition of reference potentials continue to be a challenge for all technologies in analog, digital, radio frequency design, and biomedical electronics. Many problems in system design have their origins in the difficult of establishing a common reference for measurement. So, we will start by selecting a “Ground” potential point. First, the Ground potential reference point is a circuit node or in the Place menu. We want to “Place” this node in our circuit. So, you may reach this by pointing to the **Place menu**, as in Figure 6.

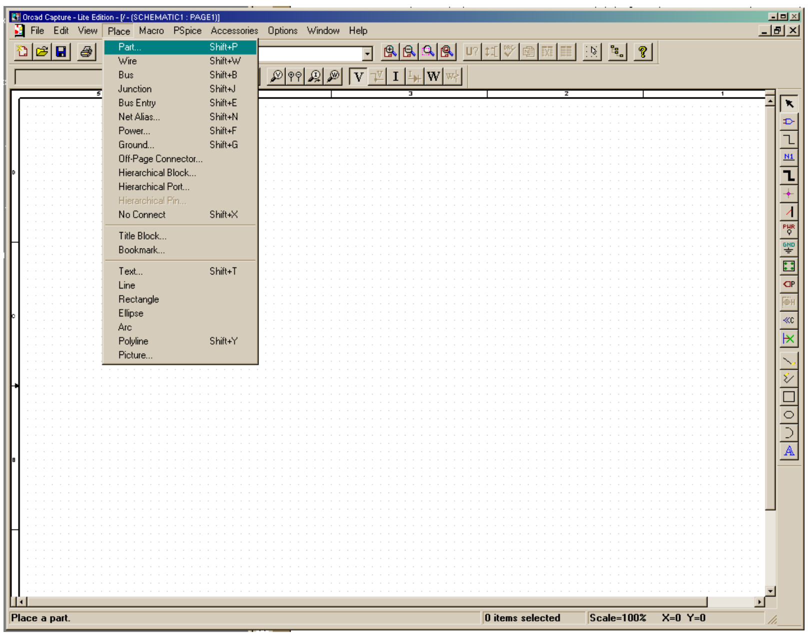

Figure 6

Now, click on **(Place -> Ground…)**, and you will see the Ground dialog box as in Figure 7.

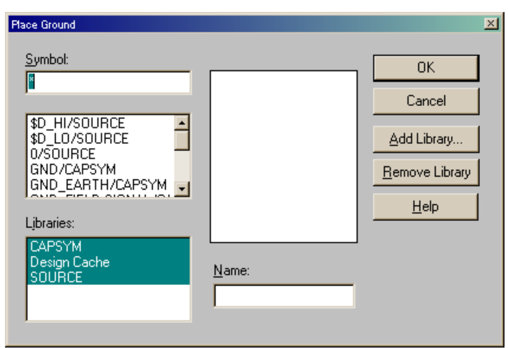

Figure 7

At this stage, we need to add Parts Libraries to our list of choices. You will need to perform this only once, in the future, the Libraries will appear by default. So, click on Add Libraries and you should see several choices.

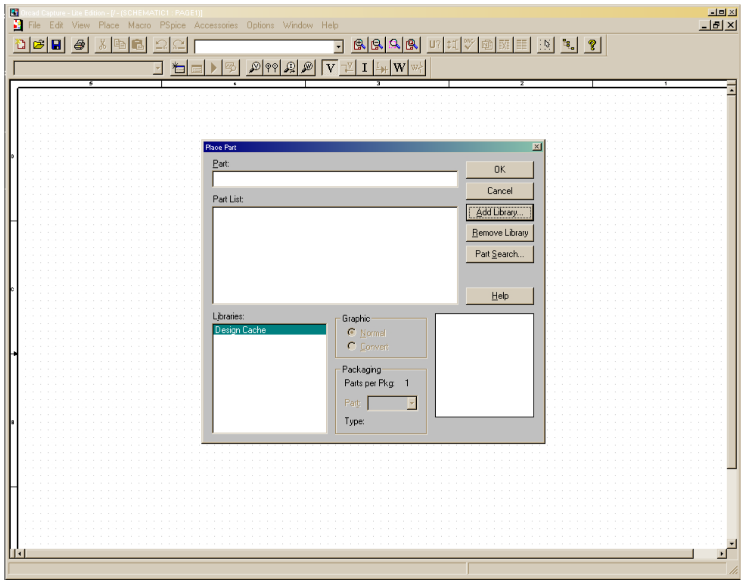

Figure 8

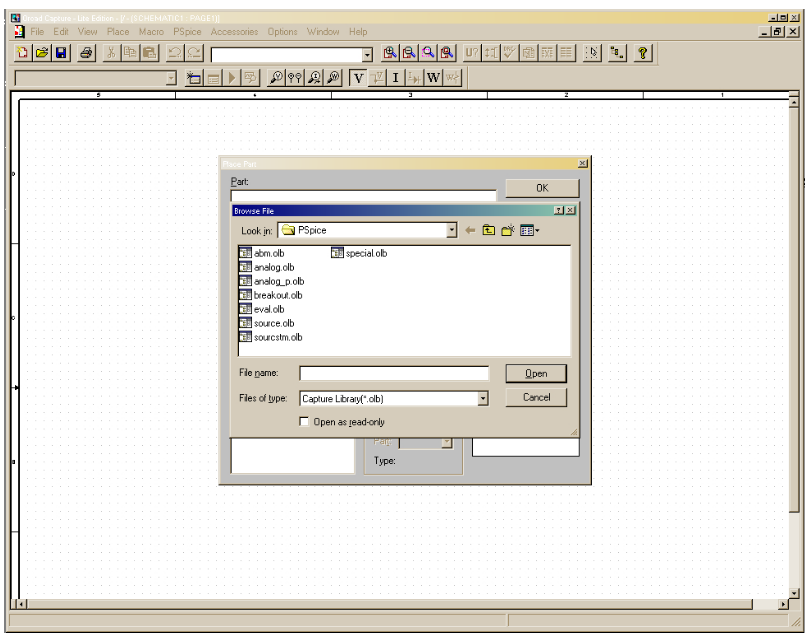

Figure 9

We need to add the following Libraries:

1) **analog.olb**
2) **eval.olb**
3) **source.olb**
4) **sourcstm.olb**
5) **special.olb**

You should see the Libraries highlighted as shown. Now, select the **0/Source** selection as in Figure 10.

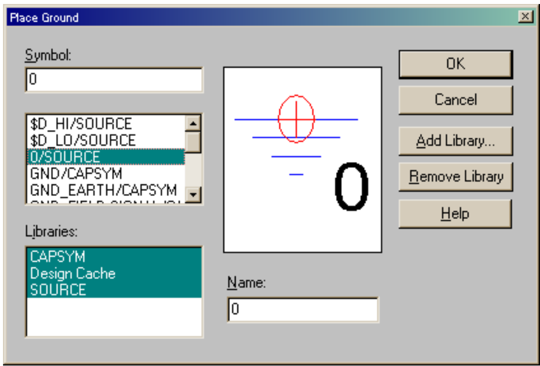

Figure 10

Then, click on **OK** and you will now find that there is a Ground symbol attached to your mouse cursor as in Figure 11.

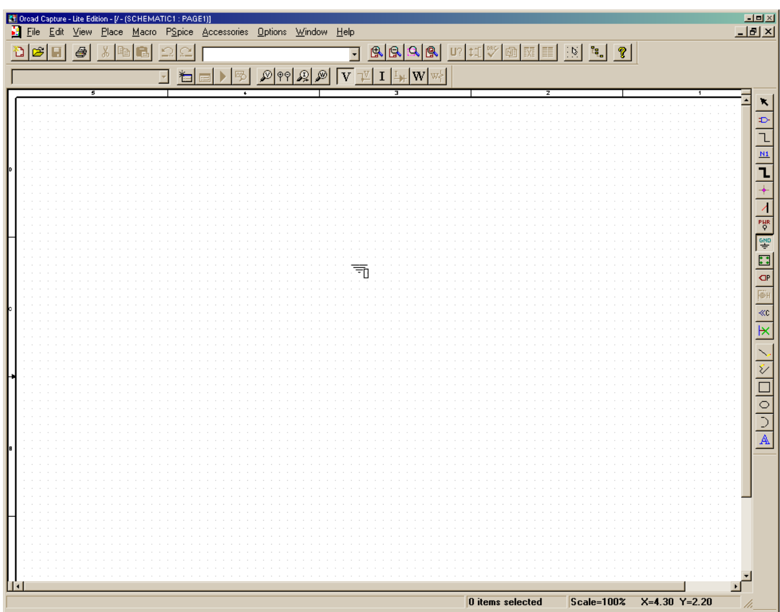

Figure 11

Place this as shown in Figure 12 by clicking with the left mouse button. Then, click with the right mouse button. If you are not careful here, you will accidentally create two ground terminals (one will have to be deleted by right clicking on the part and deleting it).

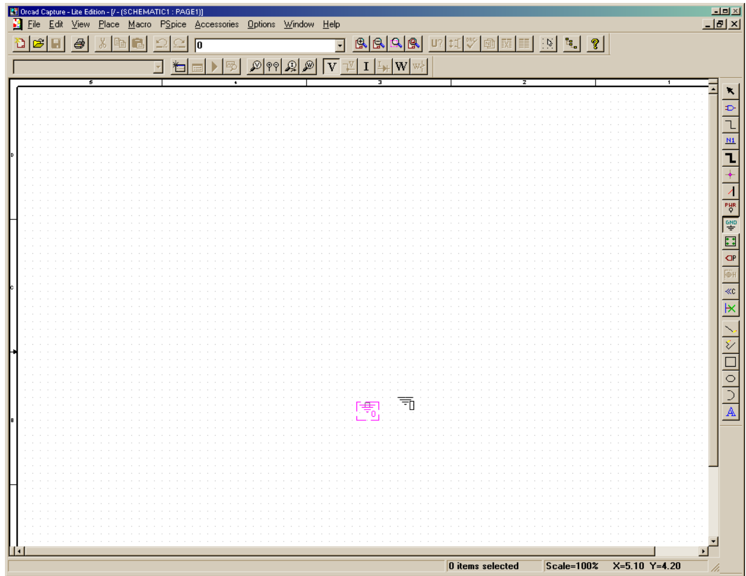

Figure 12

To view the circuit more easily, click on **View > Zoom > In.**

Now, we will add a conductor (Wire) to our circuit. Again, proceed to the Place menu, select Wire, and this will now create a drawing tool. You may create the wire by placing the mouse cursor on the terminal at the top of the Ground symbol and then holding down the left mouse button while dragging the mouse cursor upwards (as you can see this is the standard Windows drawing paradigm so this will be familiar to you.) The circuit will now appear as in Figure 13.

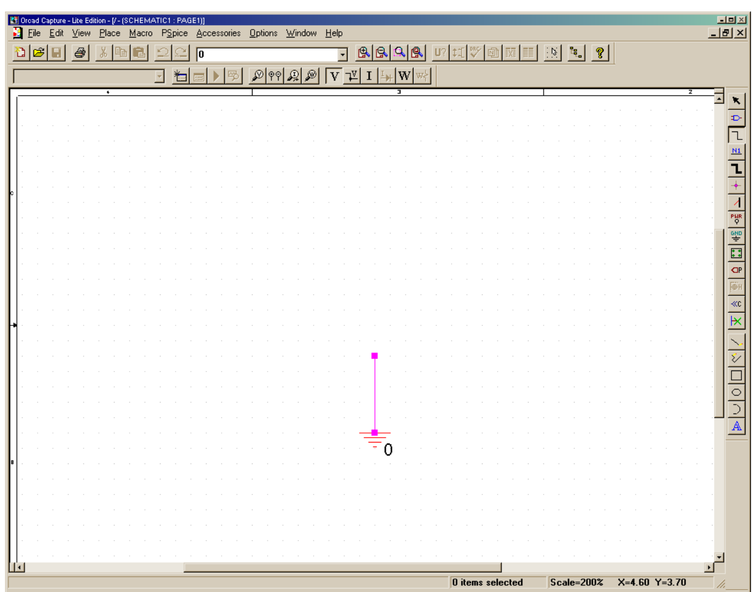

Figure 13

Now, we must start to add Resistor and Source components. To obtain and "Place" these parts we again navigate to the Place menu. But, this time, click on Part. This will bring up a Place Part dialog box. To place a Resistor in our circuit, click on **Place > Part** and then when the Place Part dialog box appears. Highlight the ANALOG library in the Libraries box. This will bring up a list of parts in the Part List box. Now, scroll down to R and click on this. You should see the dialog box of Figure 14. When we enter device property values into PSpice configuration settings or directly into SPICE code, we generally use a scientific notation for number entry that includes a scale factor equal to a power of 10. The Appendix at the end of this document describes the SPICE and PSpice Units of Measure as well as the scale factor conventions. *It is important to read this very carefully. Frequent errors in SPICE and PSpice simulation result from a confusion over entry of a proper scale factor.*

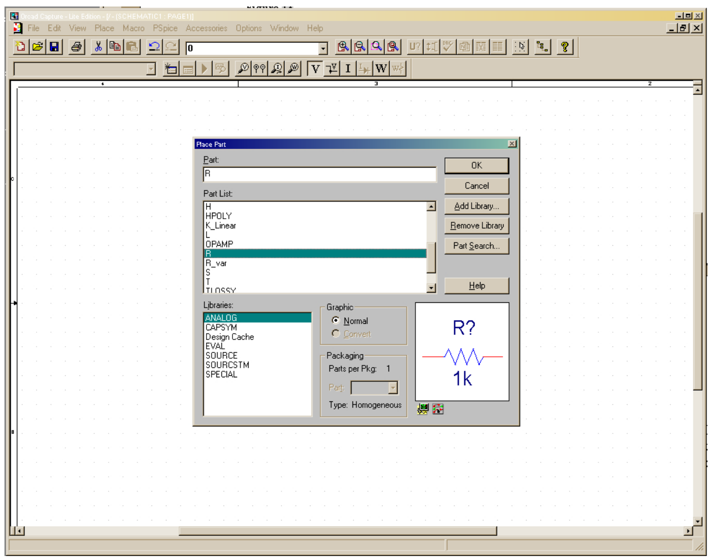

Figure 14

Click the **OK** button to select this and then you are ready to Place this part. Proceed to place this part as shown in Figure 15.

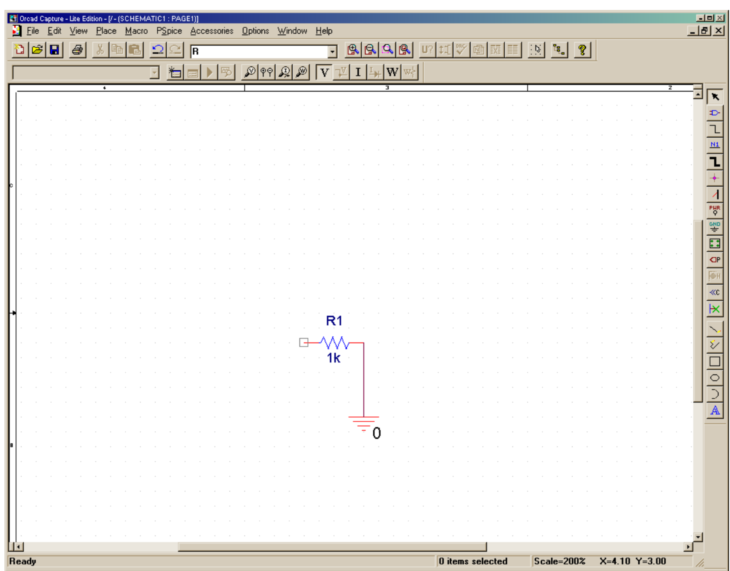

Figure 15

Its default value is 1 kilo-Ohm. We now want to change its value to 5 Ohms. So, double-click on the Part. This will bring up the Property Editor, as shown in Figure 16.

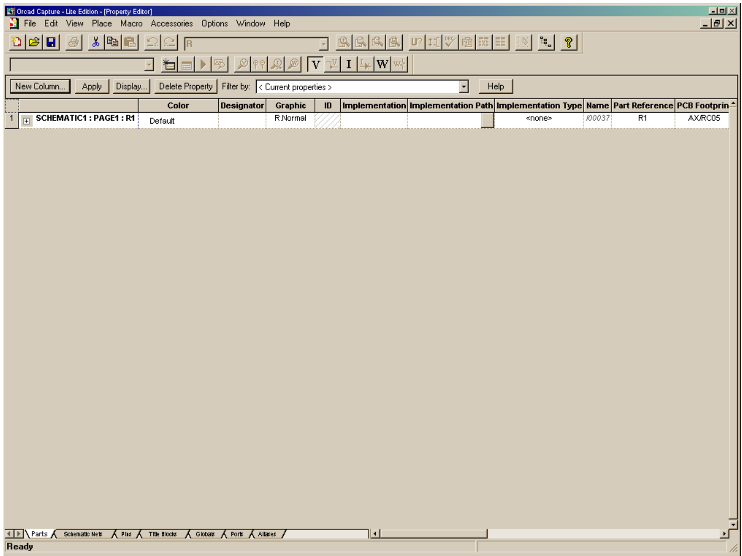

Figure 16

Now, scroll horizontally to the "Value" property box at the far right. You may highlight the contents of this box and then enter "5" to set the Resistor value at 5 Ohms.

Now, it is important to click on **Apply** at this stage to set the value. Now, return to the Schematic window via the Window Menu as in Figure 17.

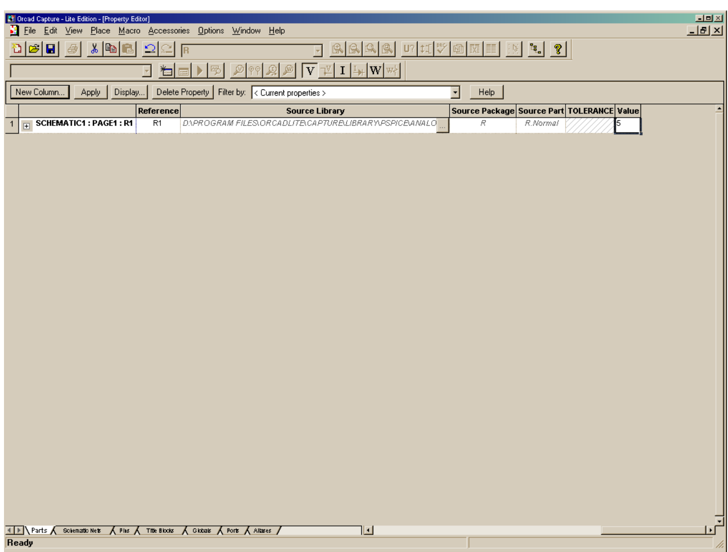

Figure 17

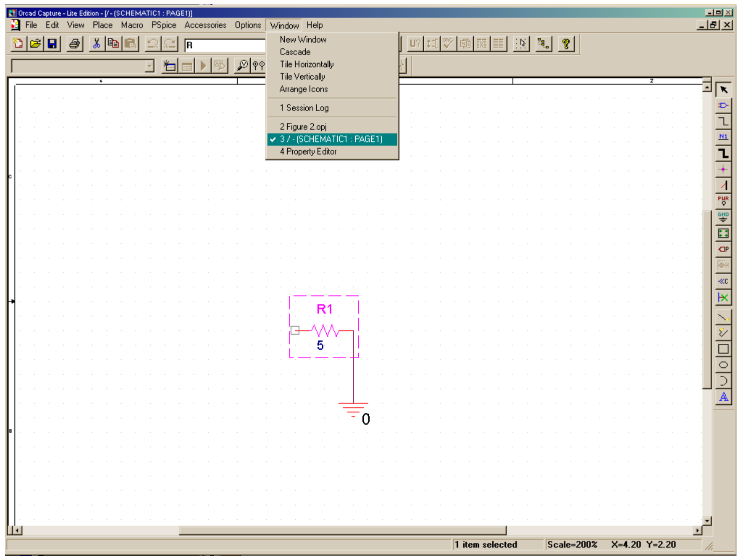

Figure 18

At this stage we will add a Current Source. This will be a direct current (DC) source and will be labeled as IDC by PSpice. So, proceed to the **Place > Part** menu to bring up the Place Part dialog. Highlight the SOURCE Library and then IDC as in Figure 19.

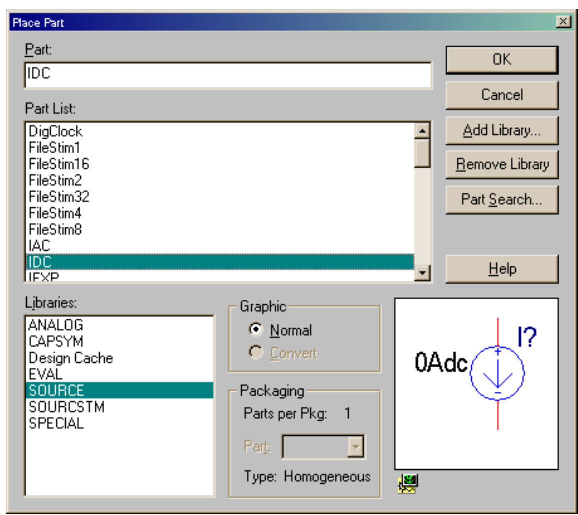

Figure 19

Now, upon clicking **OK,** we are ready to place this source. Place the Current Source so that its terminal connects to the junction of the Resistor (R1) and Ground Wire. Then, right click to bring up a configuration menu as in Figure 20.

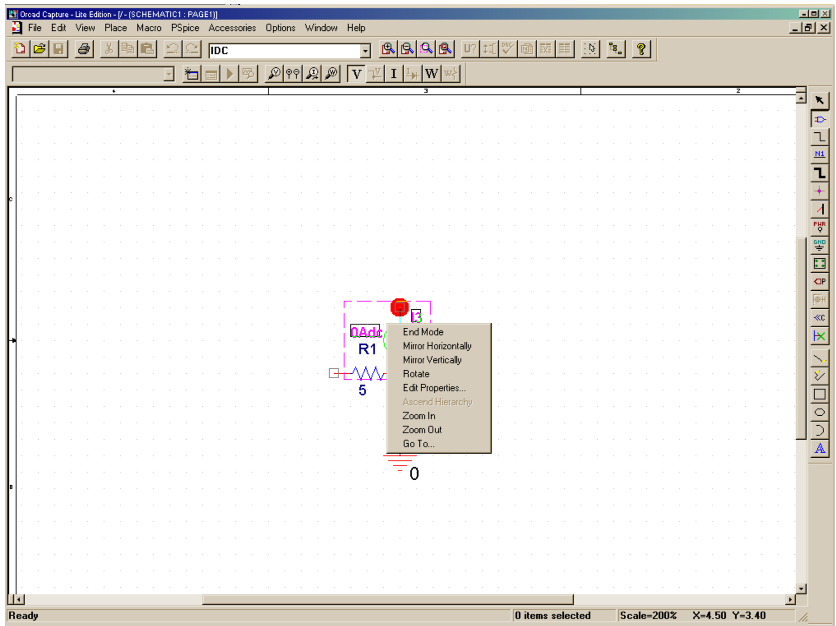

Figure 20

Now, select Rotate. You will then be able to drag the Current Source into the position shown in Figure 21. Now, use the Property Editor to set its current to 1 Ampere by updating the DC column as shown in Figure 21. (*Remember to press the **Apply** button).*

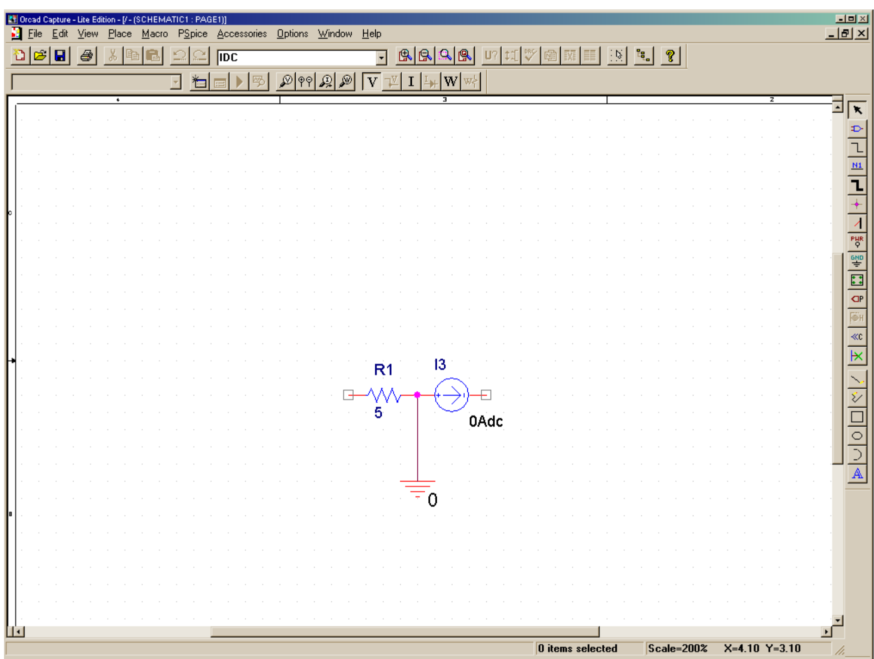

Figure 21

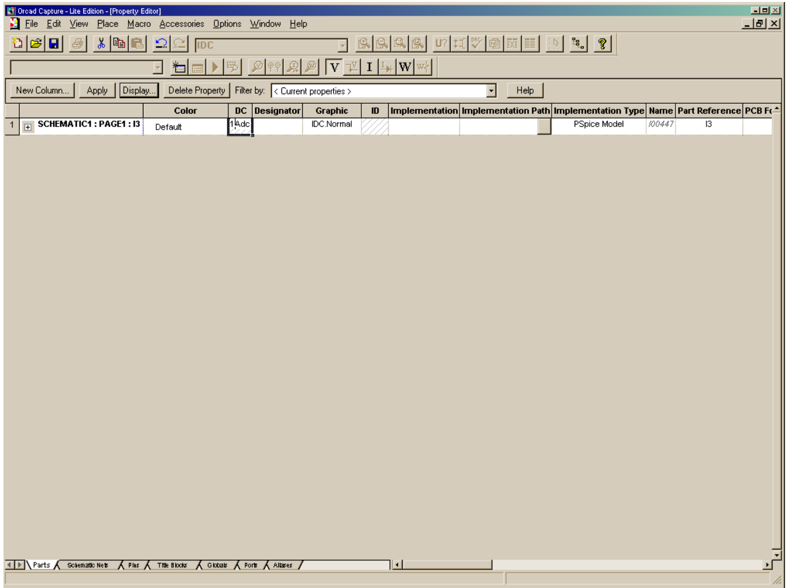

Figure 22

Now, you are ready to add additional components. All of the current sources will be set to 1 Ampere.

Here is the appearance, in Figure 23, of the circuit when additional wires, a Current Source and the 4 Ohm Resistor have been added. Note that you may click on the Part labels directly and change their values and names. Also, the labels may be moved to make a schematic more legible.

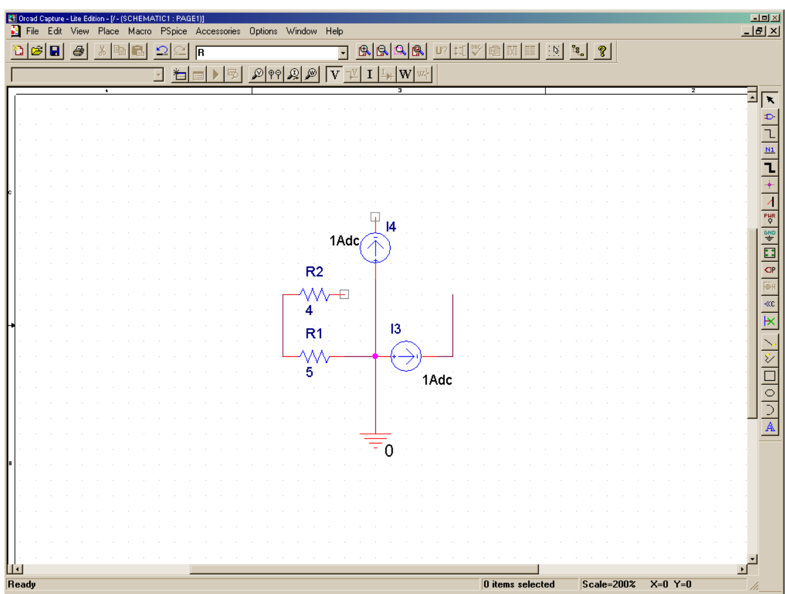

Figure 23

After additional components, we have the Schematic of Figure 24. This is a very important stage to highlight a critical feature.

Note that in the circuit of Figure 2.16 of Nilsson and Riedel, there is no connection between the conductors connecting nodes a and c and conductor connecting node d with the Current Source terminal, as shown in Figure 24, below.

We must be very careful to recognize this and to design the schematic of Figure 25 properly. Note the position of the node junctions appearing as red "dots" in Figure 25 and note that no junction appears in the center of the schematic diagram where the two conductors cross (the conductors are those connecting R2 and R3 and the conductor connecting Ground and I4).

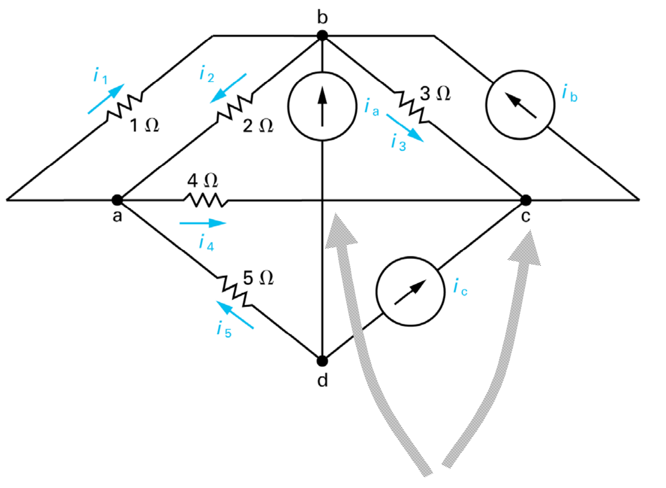

Figure 24

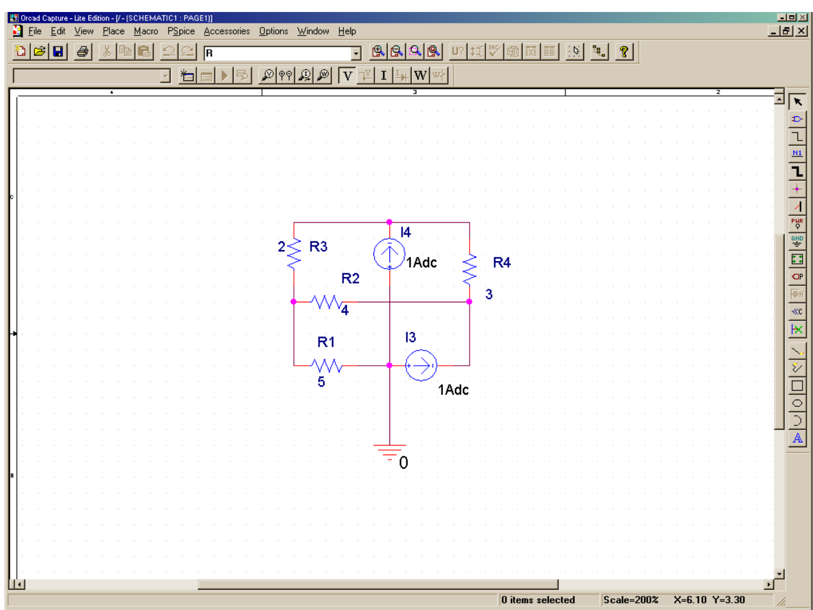

Figure 25

Now, we complete the circuit as shown in Figure 26.

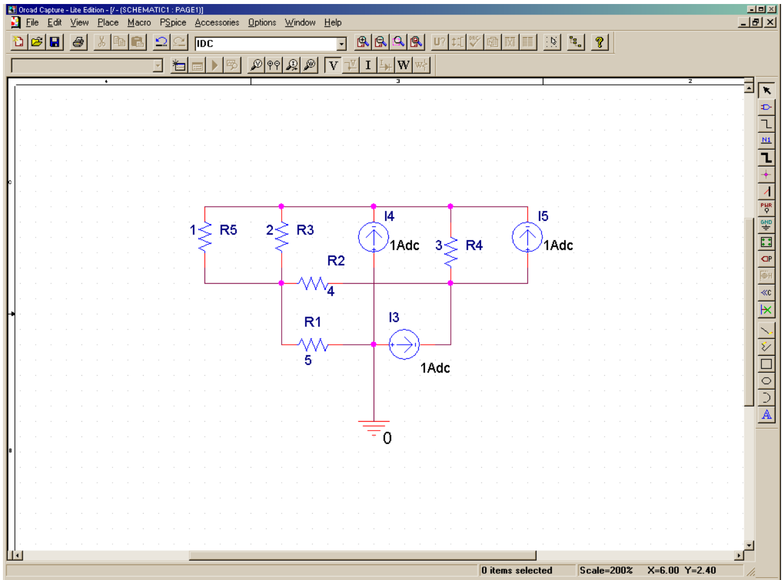

Figure 26

Now, with our circuit complete, we are ready to start a Simulation. First, we must configure the Simulation tool. Proceed to the **PSpice** menu and click on **New Simulation Profile** as in Figure 27.

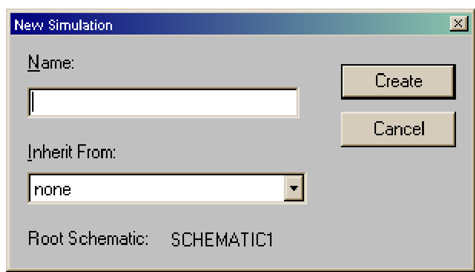

Figure 27

We have entered the name "Bias" into the **Name** field and have left the **Inherit From** entry equal to noon. As we will have discussed in lecture, the term "bias" refers to the notion of a tendency or asymmetry. Bias is a voltage generated across a device or combination of devices. It may be created directly by a voltage source, or indirectly through a combination of sources and other components.

Bias analysis refers to the operation of determining this arrangement of voltages. In general, bias analysis is critical. Later, as we encounter nonlinear circuit elements, you will see that bias analysis and design for a stable, predictable set of bias voltages is often one of the most important concerns.

So, we now click on **Create**. This will bring up the Simulation Settings-Bias dialog box as in Figure 28.

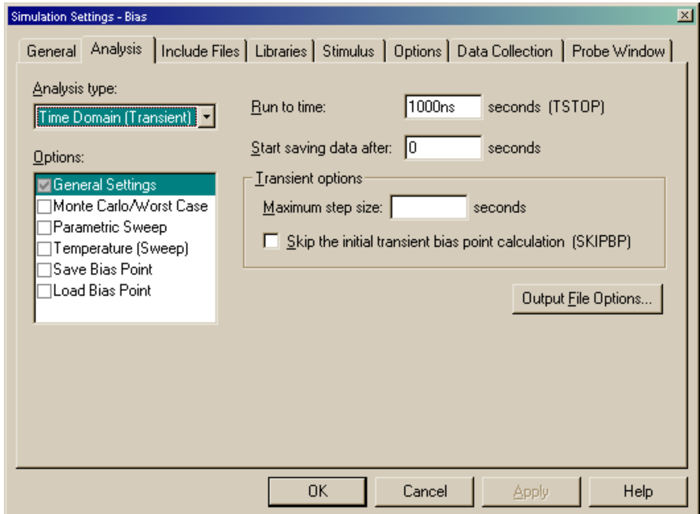

Figure 28

Now, the **Analysis type** we must select is **Bias Point.** You can reach this by scrolling through the **Analysis type** selections and clicking on **Bias Point.** We do not need to select any of the **Output File Options,** so simply click on **OK.**

Now, start the Simulation by clicking on **PSpice > Run**. The output will appear as in Figure 29.

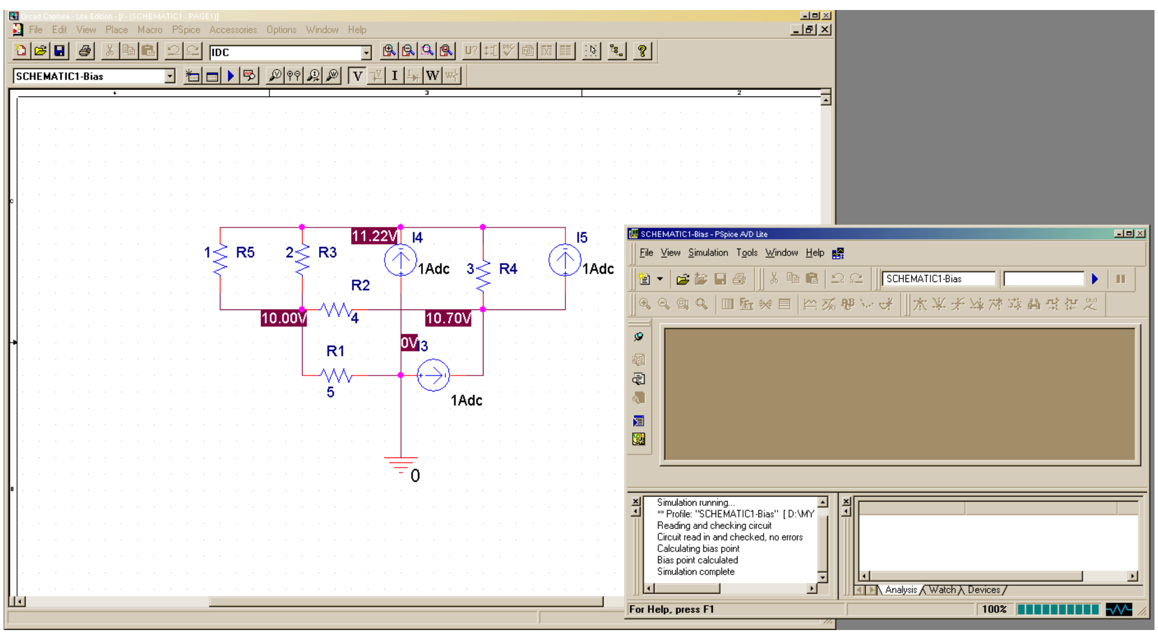

Figure 29

Note that now the circuit potentials at each node are computed. Also, we can determine the currents through each element. To obtain this information, we can examine the Simulation Output File. Navigate to **PSpice > View Output File.**

The Output File contains a PSpice program listing that was generated by "capturing" the schematic. This program listing was the target analyzed by PSpice to determine the Bias the list of bias voltages for each element, as shown in Figure 30.

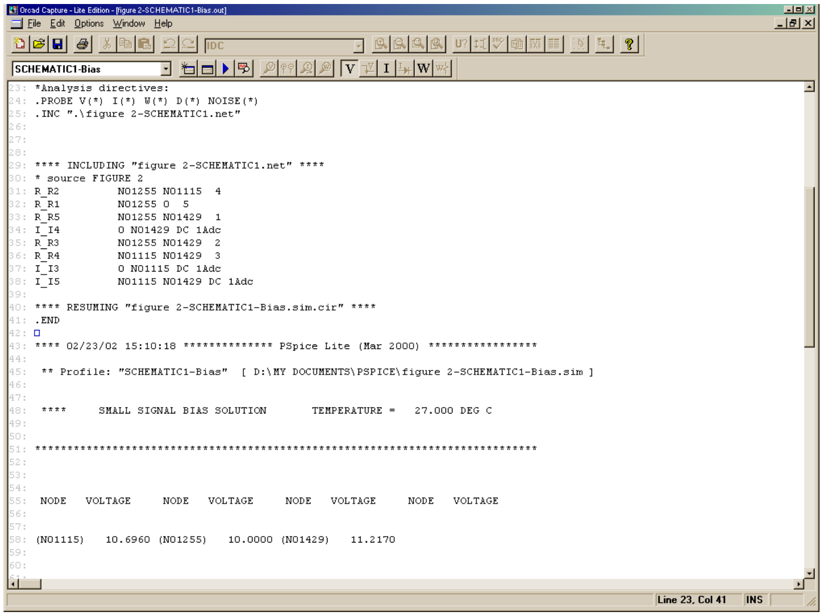

Figure 30

Also, we may examine the Bias analysis for current values. Generally, the display default will be to show Bias Voltages. To observe Currents, click on **I** and de-select **V** in the Toolbar. The Bias Current display will appear as in Figure 31.

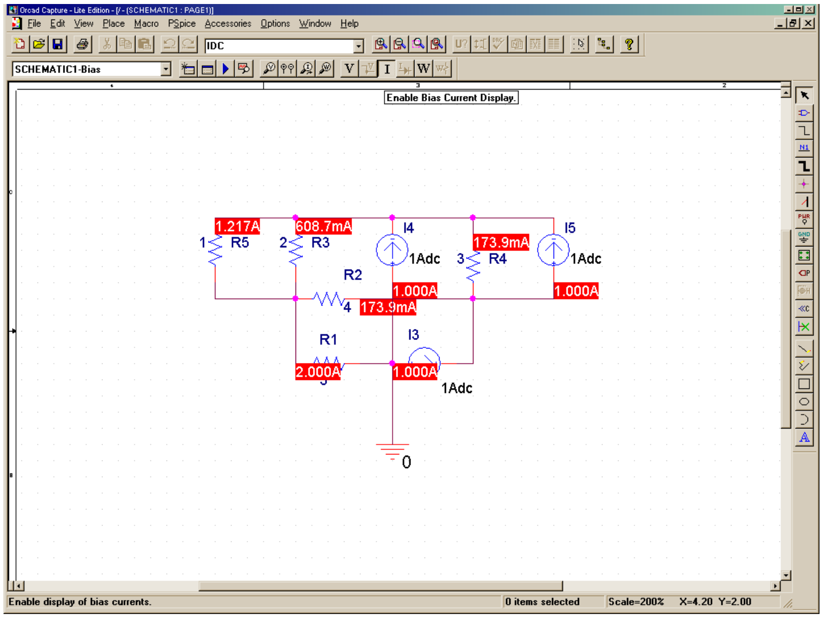

Figure 31

You may **Save** your work with the **File > Save** command. When you close your Project, you will be prompted by a pop-up dialog box to Save your Project files. You should select **Save All.**

APPENDIX:

UNITS OF MEASURE FOR SPICE

Before we proceed, however, we will discuss the units of measure that SPICE expects and will interpret from our input. As you will see, device property values in SPICE components are entered as an integer or real number, followed by a case-insensitive scale factor. For example, 1,000 Ohms may be entered as 1000, 1k, or 1K, where K is a scale factor equal to 10^3. For another example, 1,263,345 Ohms may be entered as 1.263345 meg, 1.263345 MEG, or 1263.345K, where MEG is a scale factor equal to 10^6.

If no scale factor is supplied then SPICE interprets the scale factor to be unity. Thus, if we enter 5 for a Resistor value, this will be interpreted as 5 Ohms.

The units for calculation and measurement are:

1. Resistor:	Ohm
2. Capacitor:	Farad
3. Inductor:	Henry
4. Potential:	Volt
5. Current:	Ampere

The scale factors are:

Scale Factor Term	Numerical Factor	Upper Case SPICE Factor	Lower Case Spice Factor
Tera	10^{12}	T	t
Giga	10^{9}	G	g
Mega	10^{6}	MEG	meg
Kilo	10^{3}	K	k
Unity	_	_	_
Milli	10^{-3}	M	m
Micro	10^{-6}	U	u
Nano	10^{-9}	N	n
Pico	10^{-12}	P	p
Femto	10^{-15}	F	f

Now, it is important to note that confusion can easily lead to very large errors. For example, it may be tempting to write 1 Farad as 1 F. However, this is interpreted as 1 femtoFarad and an error of 15 orders-of-magnitude will result!

Also, it is common error to use “M” or “m” to denote 1 Million, by accident. However, this is interpreted as a scale factor of Milli or 10^{-3}, an error of 9 orders-of-magnitude.

It is essential to carefully review and follow the scale factor rules. They will become familiar soon.